SHORT TURNAROUND TIME TURBULENT FLOW COMPUTATIONS FOR COMPLETE AIRCRAFT CONFIGURATIONS

J.W. van der Burg, E.T.A. van der Weide
(National Aerospace Laboratory NLR)

Keywords: *Geometry modelling, hybrid grid based CFD technology*

Abstract

Aerodynamic design and development of civil and military aircraft relies to an increasing extent on Computational Fluid Dynamics (CFD) analysis tools as time goes on. These tools are seen as complementary to wind tunnel tests and flight tests. Computational methods can provide a better insight into specific aerodynamic features. For example they can give a better insight in installation effects of geometry components, such as engine(s), flaps, slats, flap tip devices, bay doors, weapons, fuel tanks and pods on the flow field. CFD tools can be used to assess the aerodynamic characteristics of an aircraft in terms of aerodynamic coefficients such as lift, drag and moments, and to examine the behaviour of aircraft under aerodynamic high loaded conditions.

Hybrid grid technology, combining prismatic grid generation near aerodynamic surfaces with automatic tetrahedral volume grid generation, has become an important tool for aerodynamic analysis and design because it allows a higher level of automation than classical CFD.
In the two-year FASTFLO II project[1] *a viscous flow capability has been introduced in the inviscid flow based CFD system that was developed in the FASTFLO I project (Refs. 1,2,3).*

In the paper the main FASTFLO II research results will be reviewed. Application of this hybrid grid technology to high lift configurations is illustrated. Possible ways to further improve the capabilities of the hybrid grid CFD technology are discussed.

1. Requirements from aerospace industry

1.1 Motivation and potential of CFD

The European aerospace industry faces a multitude of crucial business and industrial challenges if it is to respond effectively to market opportunities arising from the continuous growth in demand for air transport. Reduced costs, faster aircraft development and reduced design risks are critical factors for competitive advantage in the changing world of aircraft design (Ref. 4).

In addition forecasts as issued by the commercial aerospace industries foresee a steady growth of air traffic and replacement of ageing aircraft over the next 20 years. To remain competitive on the international airliner market aerospace companies are under pressure to change continuously to more cost efficient development of new aircraft and derivatives. This implies that key CFD technologies for improved aerodynamic design and analysis are needed to reduce development costs and by speeding up the aircraft development cycle.

Furthermore, development and incorporation of CFD in an aerodynamic design process should be focussed on reliance (Ref. 5). High reliability is a prerequisite before CFD technology can be embedded in larger processes and high throughput is needed to explore a larger number of design concepts. Important is

---

[1] This work has been carried out in the frame of the FASTFLO II project that is funded under a contract IMT BRPR-CT97-0576 of the European Community with partners NLR (Co-ordinator), DLR, FFA, Saab, EADS, IBK and TU-Delft.

that assured quality is offered presupposing an efficient verification and validation process and user support.

1.2 Detailed industrial requirements on CFD

For CFD technology to have an impact on the aerodynamic design of aircraft the first requirement to be satisfied is that the CFD-problem-turnaround time should be in the order of a day to a week or less. Aerodynamic analysis is a process of looking at a significant number of flow conditions for more than one geometric variant, so that a large number of flow computations have to be made. If the application of CFD codes does not yield results at this industrial time scale the impact on aerodynamic design will be reduced.

A second requirement that needs to be satisfied by CFD tools for the development of commercial transport aircraft is high accuracy of predicted aerodynamic forces such that the computed aerodynamic coefficients (lift, drag, pitching moment) can be relied upon to reduce risks in aircraft design. This second requirement translates for example into better turbulence models, and extreme grid resolution or automatic adaptive grid generation if the first requirement is also to be satisfied simultaneously.

1.3 Goal of the FASTFLO projects

In view of these industrial requirements two projects have been initiated and a joint European development on a common hybrid grid based CFD technology has emerged.

The objective of the research conducted in both projects FASTFLO I (1996-1998) and FASTFLO II (1998-2000) was focussed on the development of a fully automated CFD system based on the Reynolds-averaged Navier-Stokes equations applicable to complete aircraft configurations, e.g. aircraft with engines and high lift systems.

In the first two-year project, FASTFLO I, a CFD system with an inviscid flow capability has been developed. The follow-on project, FASTFLO II, concentrated on extending the system with a viscous capability and to further increase the automation level and performance of the system. An overview of the algorithmic components of the hybrid grid based CFD system is shown in Figure 1.

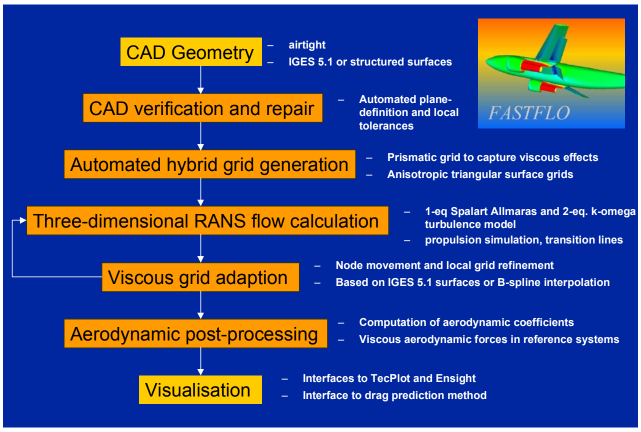

Fig. 1.Overview of the algorithm components in the hybrid grid based CFD system as developed in the FASTFLO II project. The functionality per algorithmic component is listed.

Hybrid grid technology has been selected, because it allows a higher level of automation compared to classical multi-block based CFD technology.

2 Review of the main research results of the FASTFLO II project

2.1 Overview of turbulent flow computations

In the FASTFLO II project a large number of aircraft configurations have been considered for verification and evaluation of the hybrid grid based CFD technology. An overview of the computations is shown in the Table 1.

Turbulent flow computations have been performed for both geometric components and for complete aircraft at wind tunnel conditions; Sideslip variants have not been considered.

Cases 1-6 have been selected to assess the CFD problem turnaround time. This is defined as the total working time needed (including the computing time per algorithmic component) in order to compute a viscous flow solution starting from a CFD-geometry, i.e. an airtight representation on the aircraft geometry suited for CFD purposes.

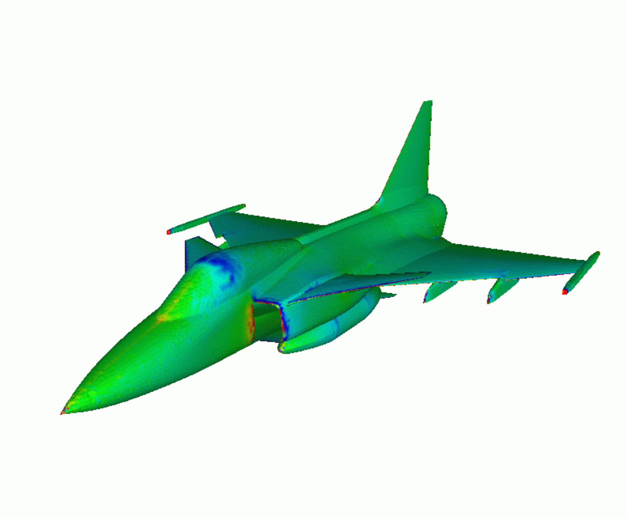

Fig. 2 Computed surface pressure for the Gripen fighter (case 1 prepared by SAAB). High pressure is shown as red whereas low pressure is shown as blue. The flow case is defined by Mach-number M=0.3 and angle of attack α=2.0 degrees.

Cases 7-13 have been selected to evaluate the CFD system with respect to accuracy. In the viscous flow computations the 1-equation turbulence model of Spalart-Allmaras (see Ref. 6) or the k-ω turbulence model (see Ref. 7) has been adopted.

2.2 Review of the FASTFLO II project objectives and requirements

Based on the outcome of the planned turbulent flow computations the following has been concluded in the FASTFLO II project.

The CFD problem-turnaround time for Reynolds-averaged Navier-Stokes computations using the hybrid grid based CFD system is within the order of one week starting from a CFD-geometry in multi-block based or IGES format for a complex aircraft configuration.

No.	Aircraft configuration	M_∞	α	Re_L ($\times 10^6$)	Transition Line	Turbulence Model	Figure
1.	SAAB Gripen fighter	0.3	2.0	n.a.	n.a.	k-ω	Fig. 2
2.	X31-baseline test aircraft	0.4	20	50	0%	SA	-
3.	X31-trimmed test aircraft	0.5	15	40	0%	SA	Fig. 3, 4
4.	X38 re-entry vehicle	0.5	10	27	5%	SA	-
5.	A3XX wing-body	0.85	fixed c_l	2.68	0%	SA	-
6.	ALVAST high-lift	0.22	12.03	n.a.	0%	SA	-
7.	RAE M2155 wing	0.806	2.5	4.1	0%	SA, k-ω	Fig. 5
8.	X31-wing	0.8	20	52	0%	SA	-
9.	DLR F4 wing-body	0.75	0.93	3.0	0%	SA	-
10.	DLR F6 wing-body-pylon	0.7	0.98	3.0	yes	SA	Fig. 6
11.	ONERA M6-wing	0.84	3.06	11.72	0%	SA	Fig. 7
12.	AS28G wing-body	0.8	2.2	11	0%	SA	-
13.	Three-dimensional cavity	0.85	0	3.45	0%	k-ω	-

Table 1 Overview of the turbulent flow computations and flow conditions as carried out in the FASTFLO II project; M_∞ denotes the freestream Mach-number, α is the angle of attack, Re_L is the Reynolds number associated to the reference length L (wbpn = wing-body-pylon-nacelle; SA = Spalart-Allmaras turbulence model).

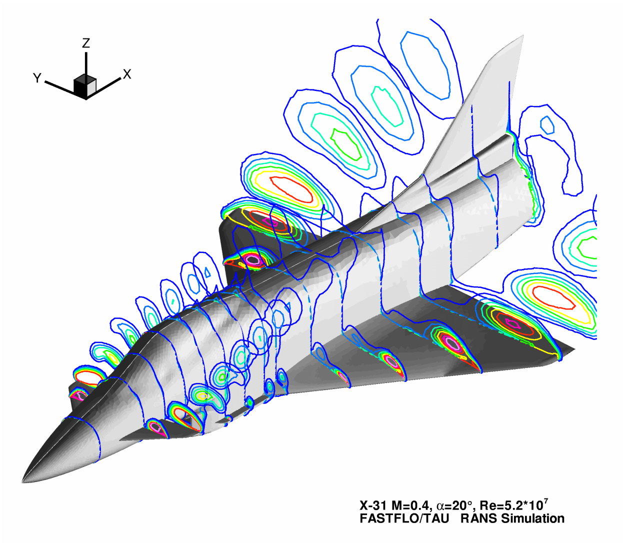

Fig. 3. Vortical structure for the X31 test aircraft at high angle of attack (case 3 prepared by EADS-M).

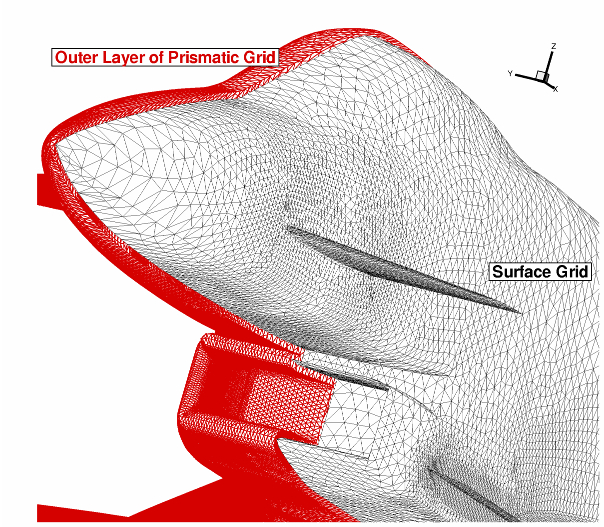

Fig. 4. Surface triangulation and the prismatic grid for the X31 aircraft (case 3 by EADS-M). Close-up of the last layer in the prismatic grid for the X31 test aircraft that has been employed to resolve the viscous boundary layer.

No.	Aircraft configuration	Turnaround time
1	SAAB Gripen fighter	4 days
2	X31 test aircraft	5 days
4	X38 re-entry vehicle	within 7 days
5	A3xx wing-body	7 days
6-9, 11-12	Wing, wing-body	1-2 days

Table 2 Turnaround time for carrying out a turbulent flow computation (including hybrid grid generation) for the indicated aircraft configurations

Turnaround times are listed in Table 2. For the X31 test aircraft one grid adaptation is incorporated in the turnaround time of 5 days. For the X38 re-entry vehicle it was concluded that the turnaround time is within one week for an experienced user.

Several critical success factors have been identified that contribute to the short turnaround time of the FASTFLO II hybrid grid based CFD-system.

First of all the grid generation process is highly automated and flexible. A limited amount of user interaction is needed to generate a viscous hybrid grid for a complete aircraft configuration.

A robust coupling to the CAD data format IGES 5.1 has been established. The coupling includes verification tests for airtightness of the CAD geometry model.

Finally, a short turnaround time is also ensured due to the high parallel scalability of the viscous flow solver algorithms. The flow solver scale linearly with the number of vector processors of a NEC SX4 supercomputer (see also Ref. 8). It is estimated that the work needed for a steady-state turbulent flow computations is in the order of $O(10^3)$ multigrid cycles.

Due to the introduction of highly automated hybrid grid generation algorithms and viscous flow solver algorithms the major workload for carrying out a viscous flow calculation has been shifted from grid generation towards CFD geometry modelling and aerodynamic post-processing. CFD geometry modelling and Computer Aided Design (CAD) data repair have become more important and more visible due to a higher level of automation in grid generation enabling CFD application to more complex geometries. This means that sufficient time and care has to be spend to decide on the geometric features and fidelity of an aerodynamic configuration.

Comparable accuracy to multi-block based technology is demonstrated for: a X31-wing, DLR F4 wing-body (Refs. 9, 10), ONERA M6 wing (Ref. 2), DLR F6 wing-body-pylon-nacelle in Figure 6, RAE 2822 airfoil (Ref. 11), L1T2 three element airfoil and the ALVAST high lift configuration. It is concluded in the FASTFLO II project that the same accuracy as in multi-block based technology can be achieved provided that a number of requirements are taken into account:

- Sufficient grid resolution (grid adaptation) and grid structure (anisotropy in both wall-normal and wall-tangential direction) is adopted. Careful tuning and design of the hybrid grid is still required.
- CFD geometry is modelled with sufficient accuracy.
- An appropriate turbulence model selected and transition line(s) specified.

Comparable accuracy to experimental data has also been demonstrated. See for instance the results of M2155 wing (Figure 5), DLR F4 wing-body (Refs. 9, 10), Aerospatiale A-airfoil, AS28G-wing-body (Ref. 2), RAE 2822 airfoil (Ref. 11).

Compared to multi-block technology the hybrid grid approach has the advantage of: a more flexible grid generation process, a high parallel efficiency, due to the application of a load-balanced grid partitioning algorithm that has no restrictions due to block-sizes and automatic local adaptation for suitable grid resolution.

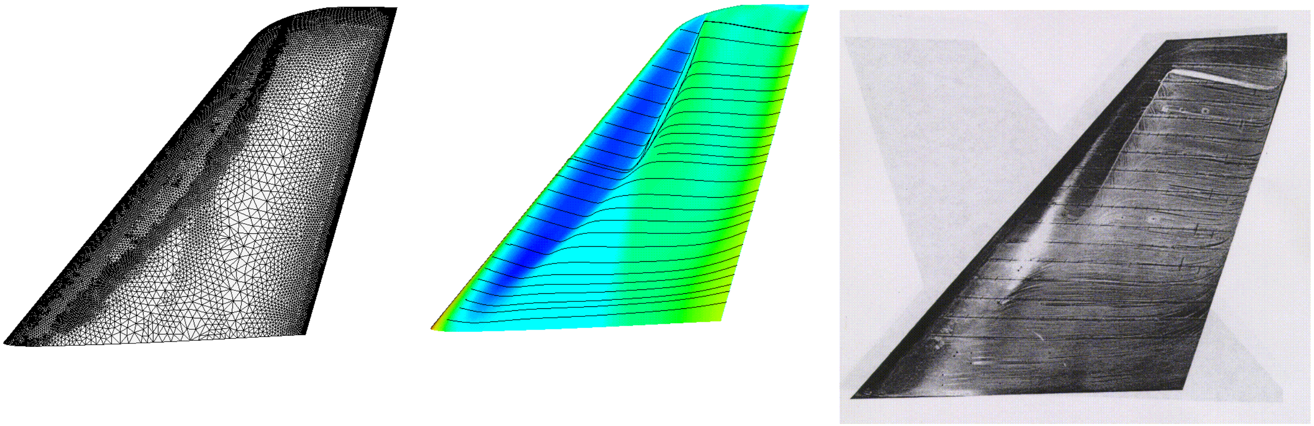

Fig. 5 Adapted triangular surface grid based on IGES 5.1 geometry description (left), turbulent flow solution (middle) and experiment (right) for the upper side of the RAE M2155 wing (case 7 by SAAB). Viscous flow computation has been performed using the k-ω turbulence model.

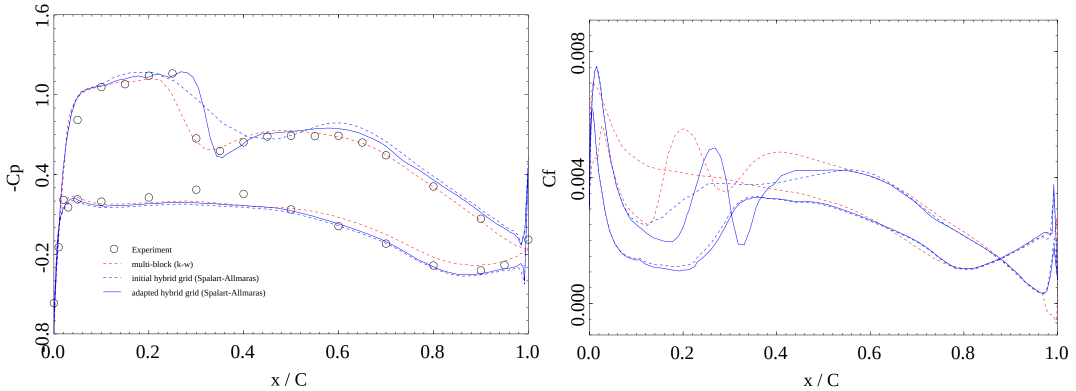

Fig. 6. DLR F6 wing-body-pylon-nacelle: pressure coefficient (left) and skin-friction coefficient (right) in span wise cut η = 0.37. Multi-block structured (Flower) results (using the k-ω turbulence model) and results on an initial and adapted hybrid grid (case 10 by DLR).

Although viscous flow results have been illustrated for many aircraft configurations, particularly for wind tunnel conditions, it should be made clear that application of hybrid grid technology to for instance high-Reynolds number flows for a complete aircraft configuration remains to be demonstrated.

The accurate computation of high Reynolds number viscous flows for example for a high-lift configuration where the flow topology becomes more complex, e.g. multiple physical phenomena occur such as transition, separation (leading edge, shock wave induced), reattachment, slip lines, requires a large number of grid points. In this context it should be mentioned as well, that similarly as for multi-block based CFD systems, turbulence modelling remains a major stumbling stone. Besides that it should be mentioned that turbulence modelling in hybrid grid based CFD systems is not yet at the same level as multi-block based methods.

Apart from improvements in flow modelling algorithms, there is also a need to further improve the grid quality. One way to further improve the accuracy is to efficiently distribute these nodes so that the respective physical phenomena are captured. To resolve the geometrical curvature, the slip lines, the finite trailing edges and the physical dominant effects poses special requirements on the distribution of the grid nodes. For example a slip line could be captured in the grid structure. Isotropic grids lead to an intolerably large number of grid nodes and therefore anisotropic grids would be beneficial needed to limit the number of nodes. Basic anisotropic hybrid grid generation has been illustrated in the FASTFLO II project. Nevertheless, the automated generation of anisotropic unstructured grids still poses a challenge.

A further extension of capabilities and integration into larger processes is under way, such as for example: development of unsteady capabilities, fluid-structure coupling, aero-elastic coupling, and interfaces to drag-prediction codes. To facilitate this integration it is necessary to reduce cost associated to the exchange of CFD-related data.

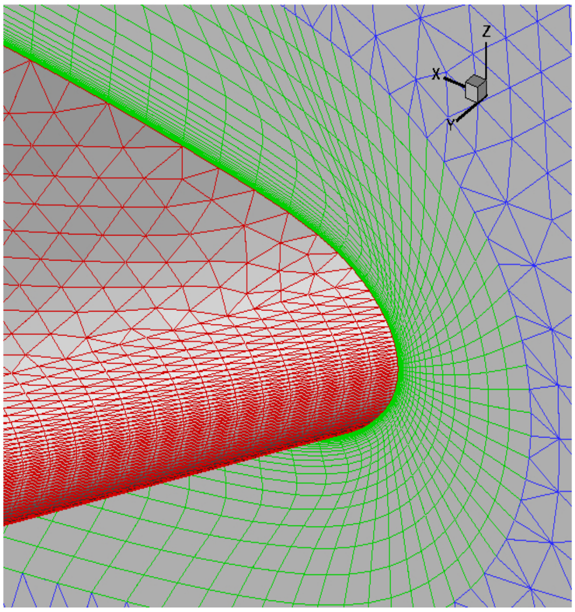

Fig. 7 Surface triangulation for the ONERA M6 wing (case 11 by NLR) including anisotropic surface triangles at the leading edge.

The specific purpose of CFD General Notation System or CGNS (Ref. 2) is to provide a standard for the exchange of CFD data associated with the numerical solution of the equations of fluid dynamics. The intent is to facilitate the exchange of CFD data between sites, between application codes, across computing platforms, and to stabilise the archiving of CFD data.

2.3 Applications.

2.3.1 High-lift configuration with engines

To illustrate the capability of the hybrid grid based CFD system a turbulent flow solution for a high-lift configuration with flaps and slats deployed and an ultra-high bypass-ratio engine installed has been carried out (see Figure 8). The turnaround time for this configuration is in the order of a week.

The total turnaround time for hybrid generation for this high lift configuration is approximately 5 hours on a SGI workstation. The hybrid grid has 30 prismatic layers to resolve the boundary layer and 7 million nodes in total in the hybrid grid, It can be observed in Figure 8 that the viscous boundary layer has been resolved: y^+ is less or equal than two.

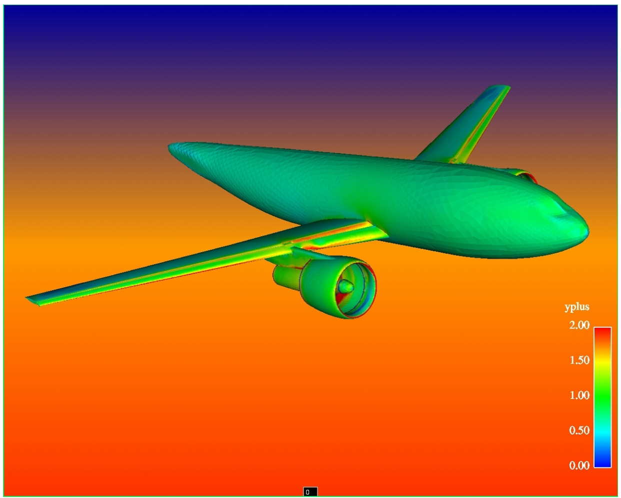

Fig. 8 y^+-distribution for a high-lift configuration with flaps and slats deployed and an Ultra High Bypass Ratio engine installed at a subsonic Mach-number M_∞=0.22 at high angle of attack α=12.03, for wind tunnel conditions Re_L=1.0 × 10^{-6}, L=0.5m

2.3.2 Eurolift

The objective of the EU research project Eurolift (Ref. 18) is to increase the understanding of the complex flow physics of high-lift systems, see figure 9, and to assess the capabilities of simulation tools like CFD.

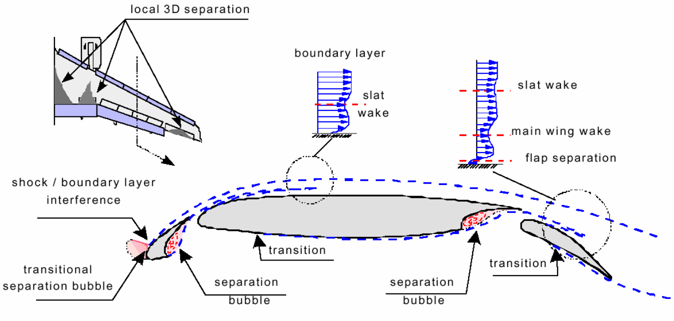

Fig. 9 Flow physics of high lift systems.

For this reason both an extensive numerical and experimental campaign, including high Reynolds numbers measurements in the cryogenic wind tunnel ETW, is carried out.

The FASTFLO system has been used to perform several computations of wing-body configurations with deployed slats and flaps. One study was to investigate the effect of flap track fairings on the high lift capabilities of a slat-wing-flap configuration. The flow conditions correspond to the high Reynolds number ETW-conditions, M_∞ = 0.2, Re = $14.0 \cdot 10^6$ (based on a chord length of 346 mm). Two hybrid grids have been created, one with and one without the flap track fairings, see figure 10 for the surface grids.

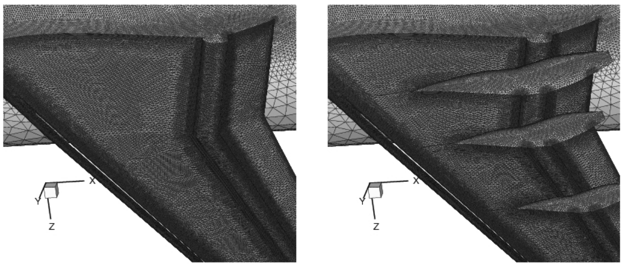

Fig. 10 Zoom of the surface grids of the slat-wing-flap configuration without and with flap track fairings.

To resolve the boundary layer 28 prismatic layers have been used with an initial normal spacing of $3.3 \cdot 10^{-4}$ mm. This spacing guarantees a y^+-distribution of one or less for the Reynolds number given. The stretching factor is variable to assure a smooth transition from the prismatic to the tetrahedral part of the grid. Both grids contain approximately 8 million points.

The pressure distribution for both configurations for α = 12° is shown in figure 11. A fully turbulent flow has been assumed and the Reynold stresses have been computed using the k-ω turbulence model. About 3000 3-level V multigrid cycles, in combination with a 3-stage explicit Runge-Kutta smoother, were needed to obtain converged solutions, which is equivalent to 100 CPU-hours on a NEC-SX5 for this grid.

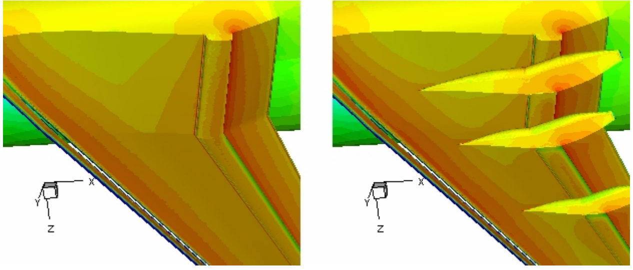

Fig. 11 Cp-distribution on the pressure side of the slat-wing-flap configuration without and with flap track fairings.

Due to the presence of the flap track fairings the pressure on the lower side of the wing is reduced, see figure 11. Consequently the lift is reduced; for this case a loss of a few percent has been observed. The presence of the flap track fairings also lead to a significantly higher drag, although numerical drag prediction for high lift configurations, especially on unstructured grids, is far from reliable yet.

The overall turn-around time for these configurations was a week.

2.4 Further enhancement of hybrid grid based CFD technology

Due to the need to reduce development time and cost of aircraft there still exists a driver to further improve the capabilities of hybrid grid based CFD technology. A further improvement of automation level is desirable to be able to explore more geometric variants of an aircraft in the same time frame. From the perspective of CFD drag prediction there is also a need to further improve the accuracy of the flow computations. Current bottlenecks are reviewed and wherever possible algorithms needed are identified.

2.4.1 CAD geometry model aspects

In the CAD geometry of an aircraft (e.g. as received from an aircraft manufacturer) many wanted and unwanted details can be present, such as for instance: finite trailing edges, small holes and gaps, small curves, small surfaces patches, sharp angled surface patches. In addition a large number curve and surface representations will be present in the CAD model. Before carrying out a viscous flow analysis these geometrical issues have to be resolved.

2.4.2 Geometric exchange

The exchange of data between commercial CAD systems is notoriously unreliable (Refs. 13, 14, 15). At this point lies a clear need for an improved standard with respect to reliable exchange of CAD-data (like the CGNS standard mentioned before).

2.4.3 Geometric analysis

In a geometry analysis of the CAD geometry aerodynamically relevant and non-relevant parts of the geometry are identified. It can be decided to locally modify the geometry and to remove unwanted small-scale geometric features. A critical issue then still remains the accurate modelling of the geometry.

Since the geometry model employed for a CFD analysis usually stems from a full scale or a wind tunnel model, the fidelity of the geometry representation is mostly not suited for immediate CFD analysis.

2.4.4 Creation of a CFD geometry

For a flow analysis a CFD-geometry needs to be prepared. This is realised by means of a CAD system. Due to the tight coupling to the IGES 5.1 CAD data format more attention can be directed towards the accurate modelling of the aircraft geometry. The tight coupling allows to inspect the geometry model more carefully, e.g. in relation to the wind tunnel geometry model utilised. Wind tunnel experiments and CFD computations should (in principle) be employed for the same geometry (without any modifications) in order to allow a good comparison.

To alleviate the airtightness requirement new CAD-coupling algorithms can be foreseen. For instance the key lines in the geometry could act as a starting point for the surface grid generation. Starting from the grid points distributed on these key-lines an advancing front algorithm would enable to grid over surface boundaries disregarding the geometry topology and geometrical irregularities (holes and gaps). This algorithm would enable to further the time spent to CFD geometry modelling since topological information is not needed any longer.

There would also be a need for a geometric analysis tool that quantifies the geometric correctness. Such a tool would contribute to a better understanding of both the CFD geometry as well as the original CAD model. As a result of such a tool bottlenecks in CFD geometry modelling could be signified earlier leading to a reduction of the time spent to obtain an accurate CFD geometry model.

2.4.5 Selection of grid resolution and topology

A CFD-specialist should be able to decide which locations of the flow domain to refine and which to coarsen. The decision on grid resolution is usually based on an "a-priori" conception of the flow topology. At the moment this is mainly a manual task. Pre-defined grid resolution could be introduced that is based on for instance surface curvature, anisotropy and the resolution of slip lines.

2.4.6 Further enhancement of the automation level of the turbulent flow solver

An often-noticed drawback with flow solvers using unstructured grids is the relative high computational cost. For a given flow condition the cost is estimated to be a factor 2-4 larger compared to the structured multi-block approach. This difference is attributed to the use of indirect addressing and a decreased efficiency due to usage of an explicit residual-smoothing algorithm. The incorporation of directional implicit residual smoothing could further improve the efficiency of the unstructured CFD technology as shown in (Refs. 16).

In those cases where large-scale flow solver computations have to be performed, however, such as for instance in CFD-studies for M-α, C_ℓ-C_d variation or in time-accurate calculations, the computational cost becomes higher compared to multi-block technology.

Nevertheless, there is evidence (Ref. 17) that an unstructured grid method can be made as efficient as multi-block based methods provided that attention is paid to data motion complexity and the reuse of data positioned in memory (e.g. for cache-based computers) near the processor(s). Extra work would be needed to improve the efficiency of the flow solver.

Choices concerning flow conditions, flow model, boundary conditions, transition location, and extent of the flow domain still have to be made. This always remains a task of the CFD-specialist. The stability of the flow calculation critically depends upon the influence of the chosen grid topology and flow topology on the underlying algorithms such as the Runge-Kutta time step algorithm (CFL-number), multigrid algorithm and the turbulence model. A flow computation requires in this respect human interaction and monitoring. The need may exist to introduce a user-friendly, intuitive graphical user interface for the use of the hybrid grid-based CFD system.

2.4.7 Grid adaptation

Concerning grid adaptation control must be asserted over the aerodynamic features that should be accurately represented such as for instance: grid redistribution for boundary layers and wakes, vorticity, surface curvature based grid generation for regions of high streamline curvature. The CFD-specialist should be able to select the features that are of interest to him.

2.4.8 Aerodynamic post-processing

In the area of aerodynamic post-processing and visualisation a trend can be observed to the usage of very large data sets. In this frame new algorithms are needed that are able efficiently reduce large amount of data (to compute aerodynamic quantities). The parallellisation paradigm can provide an outcome here to improve the level of automation.

3 Conclusions

Hybrid grid based CFD technology, combining prismatic grid generation near aerodynamic surfaces with automatic tetrahedral volume grid generation, has emerged as an important tool for aerodynamic analysis and design. Main focus of the FASTFLO II research project has been:

- To achieve a short CFD problem-turnaround time for viscous flow analysis of a complex aircraft configurations and
- To realise a high accuracy of the computed aerodynamic entities such as pressure distributions and lift, drag & moment coefficients.

Important aspect in general is that CFD-technology employed in an aerodynamic design process should have a high throughput allowing to explore a large number of design concepts. This presumes a high automation level. A short CFD problem-turnaround time for complex aircraft configurations has been realised through the judicious introduction of hybrid grid generation techniques that allow a higher level of automation in comparison with the more commonly used, conventional multi-block grid generation techniques. Through the introduction of enhanced physical modelling in the CFD system a large spectrum of fluid flow problems for complex aircraft configurations can be analysed within a short turnaround time. In this paper viscous flow analysis for high-lift configurations has been illustrated.

The accurate and efficient calculation of viscous high Reynolds number flows, however, is still a subject of current international workshops in general to evaluate multi-block structured, overset, and hybrid grids (Refs. 10,11).

A further improvement the capabilities of hybrid grid based CFD technology can be foreseen. This would also facilitate integration with other disciplines, such as fluid-structure interaction and aero-acoustics.

Large-scale application of CFD will enable aerospace industries to reduce the number of aerodynamic design cycles and create the possibility to develop innovative aero-products.

References

[1] Burg, J.W. van der; Oskam, B.; *FASTFLO Automatic CFD system for the three-dimensional flow simulations,* in: Nouvelle Revue Aeronautique, No. 2, Elsevier, April 1998.

[2] Burg, J.W. van der; Cock, K.M.J. de; Pijl, S.P. van der; *Turnaround time and accuracy evaluation of viscous flow computations on hybrid grids*, to be presented at the European Congress on Computational Methods in Applied Sciences and Engineering, ECCOMAS 2000, Barcelona, September, 2000.

[3] J.W. van der Burg, K.M.J. de Cock, T. Berglind, W. Fritz, G. Kretzschmar, N. Kroll D. Schwamborn, M. Sillen, "A joint European Initiative to develop hybrid grid based CFD technology for inviscid and viscous flow computations applicable to geometrically complex aircraft configurations", in proceedings CEAS 2002 conference, Cambridge, June, 2002.

[4] P.E. Rubbert, "CFD and the changing world of airplane design", In ICAS Proceedings, 19th Congress of the International Council of the Aeronautical Sciences, September 1994.

[5] P.G.C. Herring, "Landscape Document: An assessment of products, business drivers and aerodynamic technologies, issue number 1", Targeted Research Action in Aerospace Aerodynamics TRA3, November 1999.

[6] Spalart, P. & Allmaras, S., *A one-equation turbulence model for aerodynamic flows*, AIAA-92-0439, 1992.

[7] Menter, E. R.; *Zonal two-equation k-ω models for aerodynamic flow*, AIAA-93-2906, 1993.

[8] Galle, M.; Gerhold, T.; Evans, J.; *Parallel computation of turbulent flows around complex geometry's on hybrid grids with the DLR-tau code*, Proceedings of the 11th Parallel CFD Conference, Williamsburg, May 23-26, 1999. To appear in Verlag North Holland, Editors: Ecer A. and Emerson D.R..

[9] Gerhold, T.; Evans, J.; Efficient computation of 3D-flows for complex configurations with the DLR-TAU code using automatic adaptation, Notes on Numerical Methods, Vol. 72, Vieweg, pp. 178-175, 1999.

[10] Schwamborn, D.; Gerhold, T.; Kessler, R.; *The DLR-TAU code – an overview*, to be presented at ONERA-DLR Aerospace symposium, Paris, June, 1999.

[11] Schwamborn, D.; Gerhold, T.; Hannemann, K.; *On the validation of the DLR-tau-code*, Notes on Numerical Fluid Mechnics, Vol. 72, Vieweg, pp. 426-433, 1999.

[12] Pourier, D.M.A.; Bush, R.H.; Cosner, R.R.; Rumsey, C.L.; McCarthy, D.R.; *Advances in CGNS database standard for eachodynamics and CFD*, AIAA-2000-0681, Reno, January 2000.

[13] Wulf, A.; Steberl, R.; Akdag, V.; *Efficient integration of CFD into product design*, presented at the Industrial Computational Fluid Dynamics conference, VKI, May 1999.

[14] Brown, A.S.; *Fitting solutions for CAD-file errors*, in Aerospace America, Engineering Notebook, May, 2000.

[15] Farouki, R.T.; *Closing the gap between CAD model and downstream applications*, SIAM News, Vol. 32, No. 5, June 1999.

[16] Anderson, W.K.; Gropp, W.D.; Kaushik, D.K.; Keyes, D.E.; Smith, B.F.; *Achieving high sustained performance in an unstructured mesh CFD application,* NASA/CR-2000-210080, ICASE Report No. 2000-2, January 2000.

[17] Mavripilis, D.J.; Pirzadeh, S.; *Large-scale parallel unstructured mesh computations for 3D high-lift analysis,* AIAA-paper 99-0537, 1999.

[18] "European High Lift Programme Eurolift", Description of Work, EU Project No. GRD1-1999-10015, December, 1999.